DOCUMENT RESUME

ED 339 883 CE 059 832

AUTHOR Casey, Joe
TITLE Advanced CNC Programming (EZ-CAM). 439-366.
INSTITUTION Milwaukee Area Technical Coll., Wis.
SPONS AGENCY Wisconsin State Board of Vocational, Technical, and Adult Education, Madison.
PUB DATE 90
NOTE 57p.; For a related manual, see CE 059 831.
PUB TYPE Guides - Classroom Use - Teaching Guides (For Teacher) (052)

EDRS PRICE MF01/PC03 Plus Postage.
DESCRIPTORS Behavioral Objectives; Competence; Competency Based Education; *Computer Assisted Manufacturing; *Computer Oriented Programs; Computer Software; *Computer Software Development; *Course Content; Electronic Control; Learning Activities; Learning Modules; Machine Tools; *Numerical Control; Teaching Methods; *Technical Education; Technical Institutes; Two Year Colleges; Units of Study

ABSTRACT

This document contains two units for an advanced course in computer numerical control (CNC) for computer-aided manufacturing. It is intended to familiarize students with the principles and techniques necessary to create proper CNC programs using computer software. Each unit consists of an introduction, instructional objectives, learning materials, learning activities, and assignments. The units cover the following topics: (1) EZ-MILL, a computer software program devoted to milling operations, that students use to describe a part and to use that description to write a CNC program; and (2) EZ-EDM, a software package designed to create parts programs for electrical discharge machining (EDM). Instructional materials are illustrated with line drawings. (KC)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

09-473-150-210
WI90CD12

ED339883

439-366

Advanced CNC Programming (EZ-CAM)

U.S. DEPARTMENT OF EDUCATION
Office of Educational Research and Improvement
EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)

☑ This document has been reproduced as received from the person or organization originating it.
☐ Minor changes have been made to improve reproduction quality

• Points of view or opinions stated in this document do not necessarily represent official OERI position or policy

by Joe Casey

"PERMISSION TO REPRODUCE THIS MATERIAL HAS BEEN GRANTED BY

A Hermanek

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)."

Milwaukee Area Technical College

BEST COPY AVAILABLE

CE059832

MILWAUKEE AREA TECHNICAL COLLEGE
Syllabus
Advanced CNC Programming for Journeymen
TD MKG 439-366

1. Instructor _____ Office ____
Phone _____

2. Office Hours _____

3. Class Meets _____

4. Credits 1

5. Periods 2

6. Prerequisites CNC Programming for Journeymen

7. Textbooks and Supplies Advanced CNC Programming for Journeymen, Information and Assignment Units, (MATC) by Casey, 1 DS,HD floppy disk.

8. Course Description The purpose of this course is to familiarize the student with the principles and techniques necessary to create proper CNC programs using computer software.

9. Course Outline

Session 1	Date_____	Introduction
Session 2	Date_____	Chapter 1
Session 3	Date_____	Chapter 1
Session 4	Date_____	Chapter 1
Session 5	Date_____	Chapter 1
Session 6	Date_____	Chapter 1
Session 7	Date_____	Chapter 1
Session 8	Date_____	Chapter 1
Session 9	Date_____	MIDTERM EXAM
Session 10	Date_____	Chapter 2
Session 11	Date_____	Chapter 2
Session 12	Date_____	Chapter 2
Session 13	Date_____	Chapter 2
Session 14	Date_____	Chapter 2
Session 15	Date_____	Chapter 2
Session 16	Date_____	FINAL EXAM

10.	Assignments	Assignments for each unit will be given in class at least 1 week in advance.
11.	Requirements	Students are required to complete all assignments and tests. Attendance at every session is expected and attendance will be taken.
12.	Grading	Assignments 50%, Each Test 25%.
13.	Attendance	Attendance will be per T&I Divisional policy.
14.	Makeup Procedures	Tests must be made up in advance. 10 points will be deducted from a test score for each day that it is taken late.
15.	Dropping Courses	Per District policy as explained in the Student Handbook.
16.	Academic Alert System	Per District policy as explained in the Student Handbook.
17.	Optional References	Reference materials on CNC Programming may be found in the College Library and in the tool and die shop (T106).
18.	Bibliography	No outside reading is required for this course.

MILWAUKEE AREA TECHNICAL COLLEGE

ADVANCED CNC PROGRAMMING FOR JOURNEYMEN

Unit 1 EZ-MILL

INTRODUCTION

In this unit you will learn how to use computer software to describe a part and to use that description to generate a written CNC program. In this course we will use the Bridgeport EZ-CAM software. In this unit we will use a portion of the EZ-CAM software which is devoted to milling operations, EZ-MILL.

INSTRUCTIONAL OBJECTIVES

When you have completed this unit you will be able to:

1. Correctly enter and exit EZ-CAM software.
2. Correctly use the EZ-MILL software to create a CNC program.
3. Properly use the computer and printer to print out a part picture.
4. Properly use the computer and printer to print out a CNC program.

LEARNING MATERIALS

1. Advanced CNC Programming for Journeymen, Information and Assignment Unit 1.
2. EZ-Mill User's Manual, Chapters 1-4.

LEARNING ACTIVITIES

1. Read Advanced CNC Programming for Journeymen, Information Unit 1. (Objectives 1-4)

2. Read EZ-Mill User's Manual, Chapters 1-4. (Objectives 1-4)

3. Complete Advanced CNC Programming for Journeymen, Assignment Unit 1. (Objectives 1-4)

ASSIGNMENT

1. Read Information Unit 1.

2. Attend the Lecture/Discussions on Unit 1.

3. Read Chapters 1-4 from the EZ-Mill User's Manual.

4. Complete Assignment Unit 1.

INFORMATION UNIT 1

EZ-Mill is a software package designed to make part programming for CNC milling machines. With EZ-Mill you can define a part from a drawing using points, lines, and circles. EZ-Mill uses a series of menus to specify part geometry, tool parameters, and cycle information. It then takes that data and generates a part program.

EZ-Mill has 3 major modules. In this course we will use two of them; SIPP and POST. SIPP is used to define geometry, machining cycles, and tool paths. You can also save and recall data while in SIPP. The User's Manual has detailed explanations of all the SIPP menus in Chapter 3.

The POST module combine the part program created in SIPP with a CNC Information file to generate a CNC program for a specific machine tool.

The following steps are necessary to enter the EZ-Mill system.

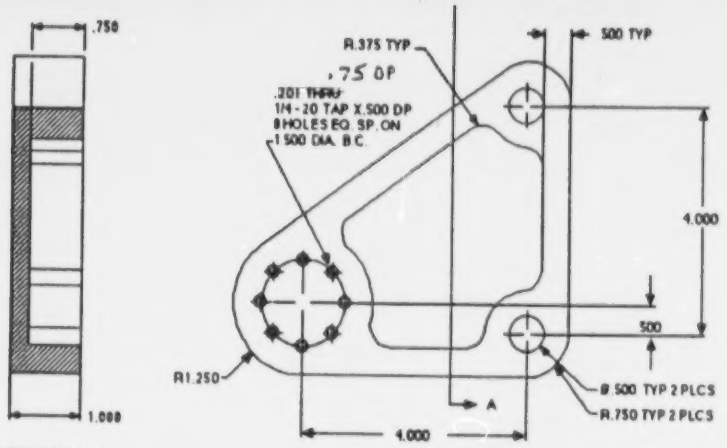
1. When the T305 Software Menu is on the screen, type EZ to enter EZ-Cam.

2. When the Windows Software comes on the screen, move the cursor via the mouse to EZ-Mill.bat and click the left mouse button twice rapidly. The hourglass symbol will appear and then you should see the EZ-Mill main menu.

We are going to use EZ-Mill to generate a CNC program for the following part. We will do this in the following order:

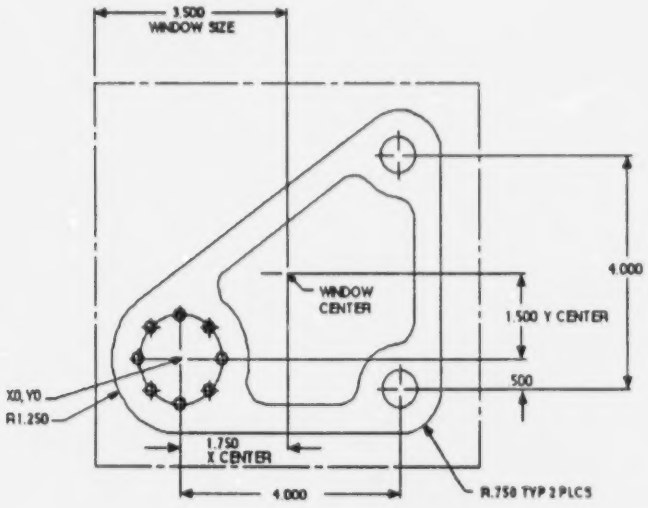
1. Rough mill the outside contour using a 1/2" end mill.
2. Finish mill the outside contour using a 1/2" end mill.
3. Rough mill the center pocket with a 1/2" end mill.
4. Finish mill the center pocket with a 1/2" end mill.
5. Drill two 1/2" diameter holes.
6. Tap-drill eight holes using a #7 drill.
7. Tap those eight holes using a 1/4-20 tap.

Refer to the following drawing as we create the program.



To begin this job you must call up the SETUP menu. Menus are called up by moving the cursor to the menu name via the mouse and clicking the mouse while the cursor is on the desired menu name. In SETUP you will determine the size and location of the working window. This established how the part will look on the screen.

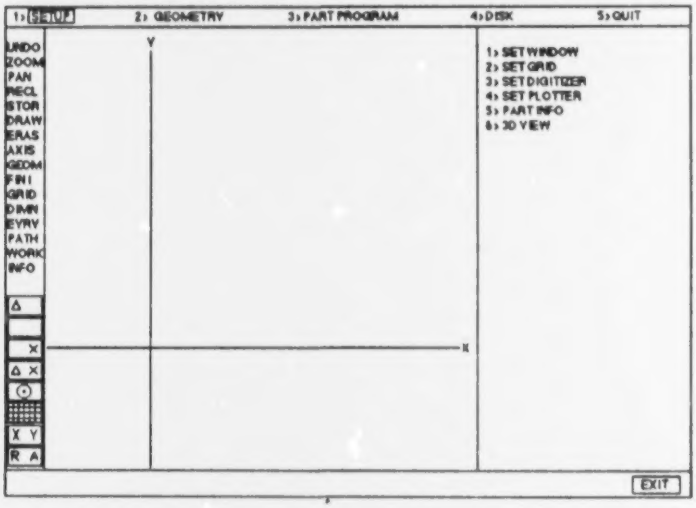
1. Decide where the 0,0 origin should be. In this job, the center of the bolt circle is 0,0.
2. Decide what the window size should be. The number you pick will be 1/2 the total width of the window. This number when doubled should be 1 inch wider than the part.
3. To set the window, call up SET WINDOW. Then enter the following data. X CENTER is the distance from 0,0 to the window center along the X axis. Y CENTER is that distance along the Y axis. SIZE is 1/2 the window width.
4. By using the next figure, you can see that we entered 1.75 for X CENTER, 1.5 for Y CENTER, and 3.5 for SIZE.



More information on the SETUP menus can be found in the User's Manual, Chapter 3, pages 13-20.

The following steps are necessary to correctly set and store this window:

1. Choose SET WINDOW.
2. Choose the AXIS command from the list of concurrent commands running down the left of the screen. More information on these commands can be found in the User's Manual, Chapter 3, pages 3-7. This command will display the X and Y axis as yellow lines.
3. Choose X CENTER and type in 1.75 in the blue numeric entry box in the lower right corner of the screen. Press the ENTER key when done typing. This key must always be used after typing in an entry.
4. Select Y CENTER and type in 1.5 and ENTER.
5. Select SIZE and enter 3.5.
6. This step is critical to storing a window size! Move the cursor the EXIT box at the right bottom of the screen. Click the mouse on EXIT. This returns you to the SETUP menu and causes the shift in window size and location to occur. The screen should look like the following figure.
7. To store the change, select STOR from the concurrent commands using the mouse and click anywhere above the YES/NO line to say yes.



To create the geometry necessary to describe the part do the following steps:

1. Select GEOMETRY from the SIPP top menu. Notice that DEFINE is the default mode for GEOMETRY. There are 9 modes and they are explained in the User's Manual, Chapter 3, pages 24-84. The DEFINE mode is used most often and it is explained in detail on pages 28-58 in Chapter 3 of the User's Manual.

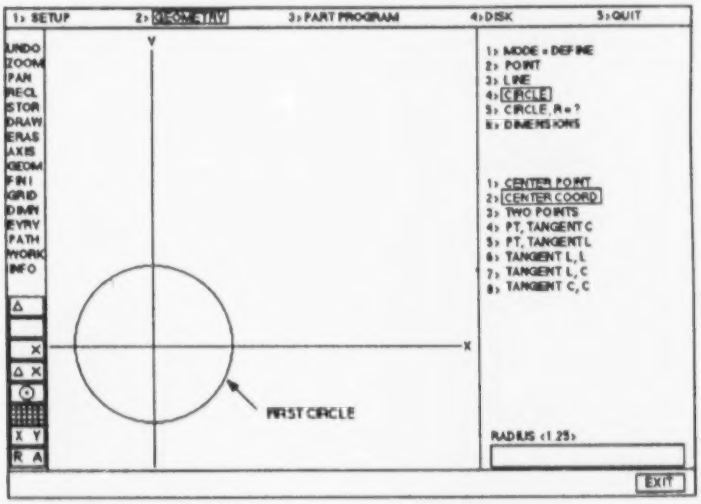
2. Select CIRCLE from the list of types of GEOMETRY. This will allow us to define the 3 corners of the piece.

3. Under CIRCLE choose CENTER COORD. This allows us to put a circle of a certain radius at a X,Y intersection point.

4. Type in 1.25 for the RADIUS of the first circle and press ENTER.

5. Because this circle, the lower left hand corner of the part, has its center at 0,0 we can just press ENTER for both X and Y positions. X0 and Y0 will come up automatically unless we change them.

6. The circle should now appear on the screen after we press ENTER for the Y position. See the following figure. If the circle is incorrect, use the UNDO concurrent command.

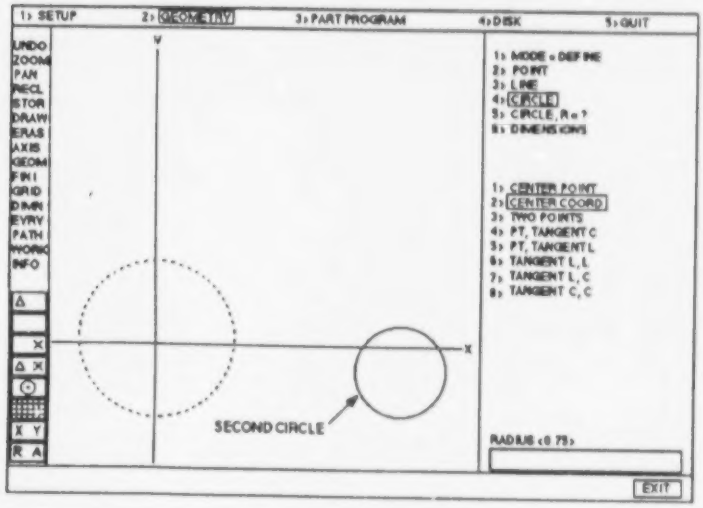


7. To do the lower right circle, type .75 for RADIUS and ENTER.

8. For CENTER X enter 4.

9. For CENTER Y enter -.5. These two numbers refer to the center of that radius which is 4 inches to the right of (+X) and .5 inches below (-Y) our 0,0 point.

The second circle should appear and look like this:

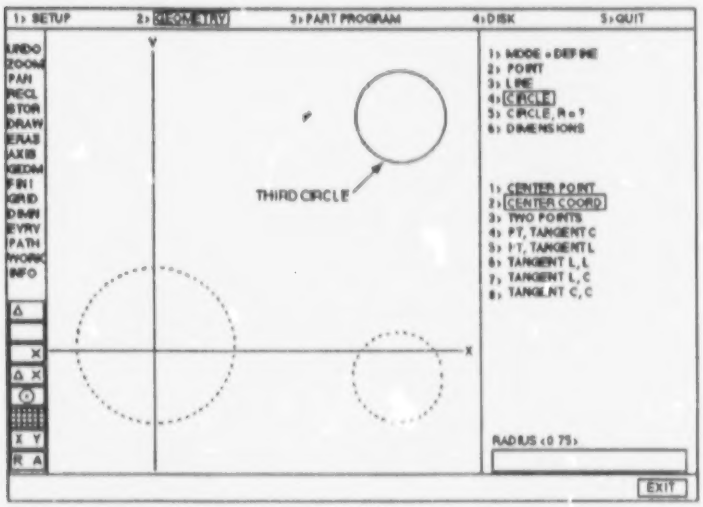


For the third circle:

10. Press ENTER for the radius because .75 is already there.

11. Press ENTER for CENTER X because the default value of 4 is correct.

12. Type 3.5 for CENTER Y and ENTER.



Now we will put the lines that from the outside of the piece on the screen using the following steps:

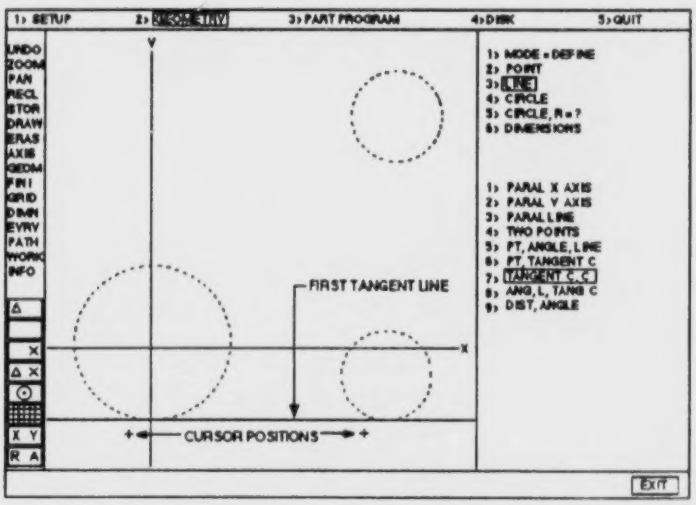
13. Choose LINE from the GEOMETRY menu.

14. Pick TANGENT C,C from the LINE choices. This will define a line that is tangent to two circles.

15. When the prompt PICK CIRCLE WITH CURSOR appears at the bottom of the screen, click just below the largest circle. That circle will turn red.

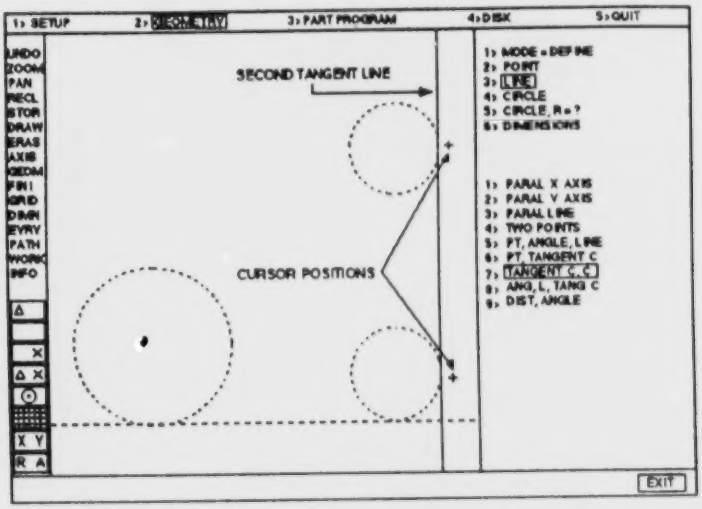
16. Now click just below the lower right circle. A green line will appear which is tangent to those circles. If the cursor is not clicked on the correct side of the circle, the tangent lines could appear but on the wrong side of the circles in question.

Your screen should now look like this:



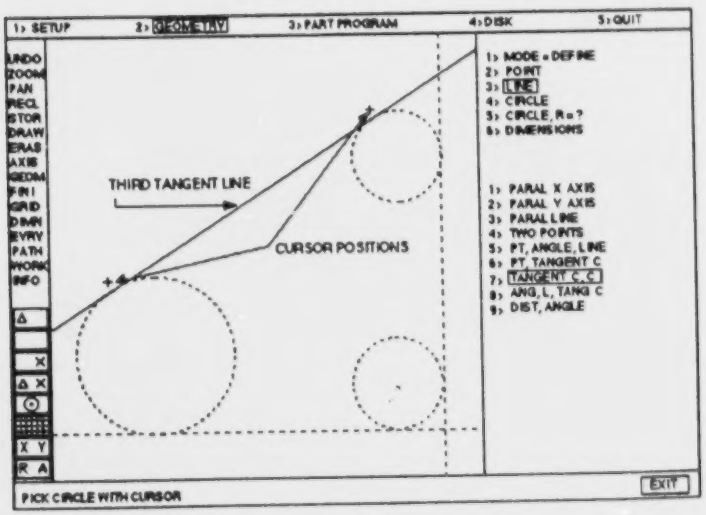
17. To do the right vertical tangent line, click to the right of the lower .75 radius circle and then click to the right of the upper .75 radius circle. This will create a vertical tangent line.

The screen should look like this:



18. To define the angled tangent line, click to the left of the upper .75 radius circle and then click to the left of the large circle. The angled tangent line should appear.

Your screen should now look like this:



There are many other ways to define lines and circles. These are explained in detail in the User's Manual, Chapter 3, pages 33-55.

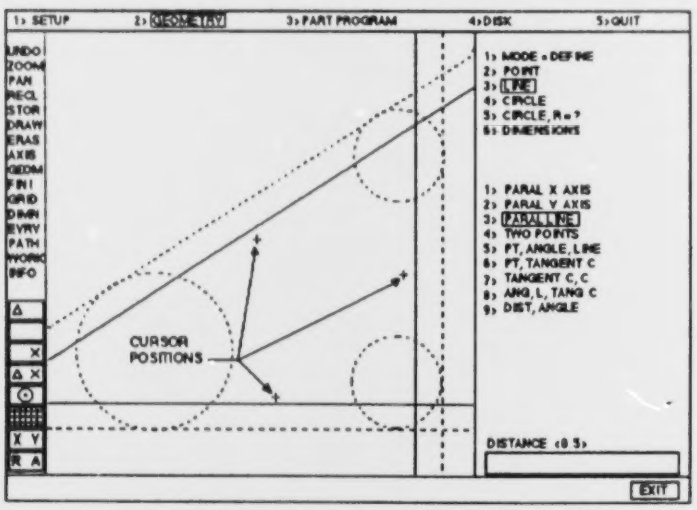
We will now create the geometry for the inside pocket using the following steps:

19. Choose PARAL LINE from the choices under the LINE menu.

20. Type .5 in the numeric entry box and ENTER.

21. In response to the PICK LINE WITH CURSOR prompt select all three of the existing lines, one at a time, by clicking on the inside of each of the lines. This tells the computer to place a line parallel to each of these lines at a distance of .5 inch and to the inside of the existing line. If we click on the outside of the existing lines, a parallel line will be created .5 inch to the outside of the existing line. By doing this, we have created the inside pocket geometry.

The screen should look like this:



Believe it or not, we have defined all of the geometry needed by EZ-Mill to complete a CNC program! We are not done however. Many other questions need to be answered. We will now use the third option from the SIPP top menu, PART PROGRAM. PART PROGRAM is explained in depth in the User's Manual, Chapter 3, pages 85-121.

To set the parameters for this piece complete the following steps:

22. Select PART PROGRAM from the top menu.

23. Choose SEG ID. A prompt will appear at the bottom of the screen asking for a SEGMENT ID. It is very important to ID every segment with a number followed by a dash followed by a short descriptive phrase. If this is done correctly, the computer recognizes the segment correctly and they may be edited easily. Each and every machining segment must have its own segment ID. On page 3 we listed 7 machining operations and we will have 7 different segment IDs. Type and ENTER 1-PROFILE for the first segment ID.

24. The MILLING CYCLE (Y/N) prompt will appear. Click above the Y/N line.

25. Select CONTOUR MILL because we are going to machine the outside contour of the piece.

26. Choose SEGM PARAMS.

27. Choose EFF TOOL DIA and type in .5 for the 1/2" end mill. (There are many menu choices that are available. In this tutorial we will use only the ones that we need. The reader is referred to Chapter 3 of the User's Manual for further explanation.)

28. Select TL CHG (X) and type in the X tool change position given to you by the instructor.

29. Choose TL CHG (Y) and do the same.

30. Select COOLANT and toggle through the choices to MIST.

31. Select FEEDS & SPEEDS from the SEGM PARAMS menu.

32. Pick SPINDLE SPEED and type in 3000. This sets the spindle at 3000 RPM.

33. Choose Z DATA form SEGM PARAMS menu.

34. Select Z DEPTH and type in 1 to establish the total milling depth at 1 inch. This is per the part drawing.

35. Select Z STEP and enter .25. This will cause the milling machine to mill the 1 inch total depth in 4,.25" steps.

36. Choose CYCLE DATA from the PART PROGRAM menu.

37. Select OFFSET VALUE. Type in .25+.025 and hit enter. Notice that the computer will do that math for you and displays the value of .275. The .25 is the cutter radius and the .025 is the finish allowance.

38. Choose CREATE PATH from the PART PROGRAM menu.

39. Select DEFINE.

40. Select ARC TANG from the DEFINE menu. There are a variety of ways to define a path. ARC TANG works best with a combination of tangent lines and arcs. See pages 106-111 in Chapter 3.

41. Click on the X icon. The icons are below the concurrent commands on the left of the screen. They define how points are to be selected. An explanation of all of the icons can be found in the User's Manual, Chapter 3, pages 7-9.

42. When you see the PICK INT POINT WITH CURSOR prompt, start defining the path by moving to and clicking on each intersection of lines (circles tangent to lines are considered intersections at the point of tangency). Do this in a clockwise direction around the outside of the part. The direction is very important because it determines whether the machine climb mills or conventional mills the piece. A clockwise direction around the outside of the piece will be OFFSET LEFT, counterclockwise will be OFFSET RIGHT. On an inside pocket, clockwise is OFFSET RIGHT and counterclockwise is OFFSET LEFT.

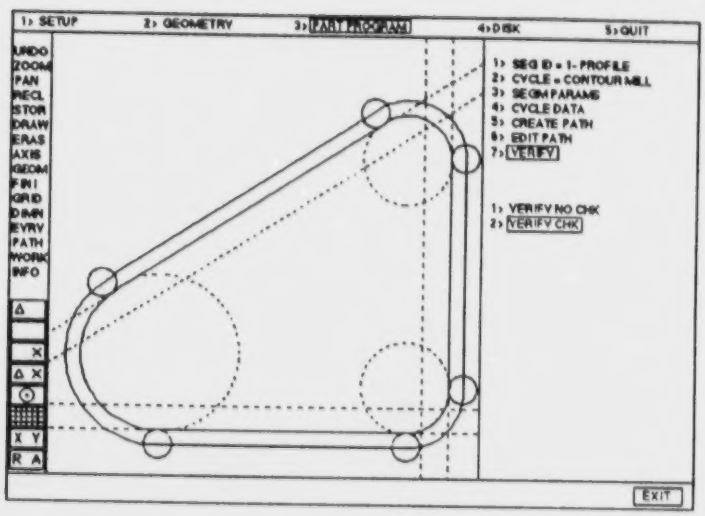
43. Notice that little triangles are appearing at each intersection. These are called points. You must complete the path with a point. Select the POINT icon before completing the last segment of the blue lines and then click on the point that appeared at the first intersection. You must end a path with the POINT icon.

We can visually simulate what we have told the computer to do at the end of any segment. It is very important to do this because it allows us to catch any mistakes. To do this:

44. Select VERIFY from the PART PROGRAM menu.

45. Choose VERIFY CHK. The COMPUTING message will appear and then a yellow spindle center line with a red cutter diameter should appear illustrating the tool path. If there are any mistakes, go to the menu choice that is necessary and change it. The tool path does not have to be redone! If it is correct, choose the concurrent command DRAW to erase and redraw the screen without any verification graphics.

Your first verification should look like this:



Our next segment will be to finish mill the outside of the piece. To do this we must change the data in the following categories; Z DATA and CYCLE DATA.

46. Select SEG ID, at the prompt type in 2-FINISH and ENTER.

47. Say yes to MILLING CYCLE? (Y/N).

48. Select CONTOUR MILL.

49. Choose Z DATA from the SEGM PARAMS menu.

50. Select Z STEP, enter 0. This 0 Z STEP will mill the entire outside of the piece in the Z axis and eliminate the cutter marks left by the Z steps in the roughing cycle.

51. Select CYCLE DATA.

52. Choose OFFSET VALUE, type and enter .25. This is the cutter radius and this setting moves the edge of the cutter in place against the finish wall of the piece.

Since this tool path should follow the roughing path, we do not have to create a new path. We will copy the path from segment 2.

53. Pick CREATE PATH.

54. Now choose COPY.

55. The SEGMENT ID prompt will appear. The computer wants to know which segment to copy. Type 1 because we are copying the tool path created in Segment 1. The message PATH COPIED will appear.

Verify the tool movement for segment 2 following the verification steps used in Segment 1. In Segment 3 we will rough out the pocket using the same 1/2" end mill.

56. Choose SEG ID.

57. At the SEGMENT ID prompt, type 3-POCKET and enter.

58. Say YES to MILLING CYCLE?.

59. Choose POCKET MILL. This will give us a different list of responses because we are now milling out a pocket instead of following a contour.

60. Choose SEGM PARAMs.

61. Select Z DATA.

62. Pick ZDEPTH and change it to .725. The pocket should be .75 deep minus .025 finish allowance on the pocket bottom.

63. Pick ZSTEP and enter .25.

64. Choose CYCLE DATA and then set the OFFSET VALUE at .275. This is cutter radius plus .025 wall finish allowance.

65. Set the STEP OVER at .30. This is how much the cutter steps over to continue milling out the pocket. The rule for stepover is 60% of cutter diameter.

66. Set the PLUNGE DISTANCE at .25. Plunge distance is how far away from the wall the cutter moves to begin its ramped radius down to the specified Z depth. The ramped radial move allows a non-center cutting end mill to be used.

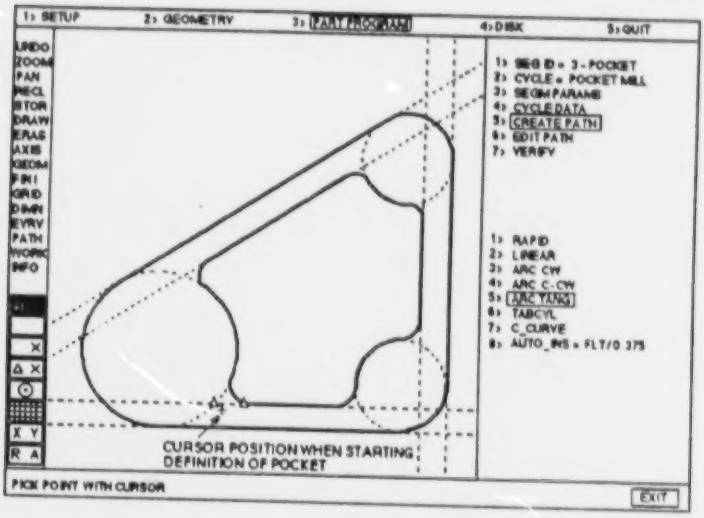
67. Select CREATE PATH and choose DEFINE.

68. Select the AUTO_INS option. This allows us to automatically insert either a chamfer or a radius of a specified size as we create the tool path.

69. Choose FILLET and set the size at .375.

70. Select ARC CW and the X icon.

71. At the PICK INT POINT WITH CURSOR prompt, move the cursor from intersection to intersection in a counter-clockwise direction around the perimeter of the pocket geometry. Start with the intersection of the 1.25 radius circle and the horizontal line. Use the following figure as an example.

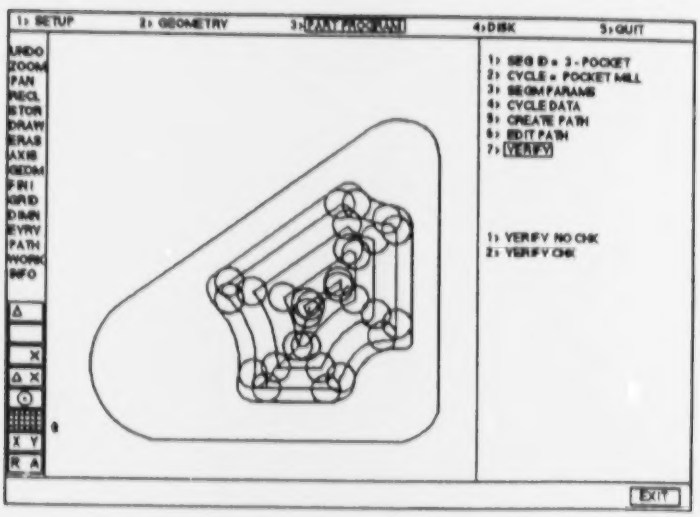


If you wish to see a blue tool path on the screen without seeing any geometry, simply choose ERAS from the concurrent commands on the left to erase the current screen and then choose EVRY from the concurrent commands to redraw all existing tool paths.

72. Select VERIFY.

73. Pick the VERIFY CHK option.

74. After the COMPUTING message appears, the verification should look like this. Pick DRAW to erase and redraw the screen after the verification is done.



Now we will finish mill the pocket.

75. Choose SEGMENT ID and enter 4-FINPOCKET at the prompt.

76. Answer YES to MILLING CYCLE?.

77. Select POCKET MILL.

78. Choose SEGM PARAMS and select the Z DATA option.

79. Select ZDEPTH and change it to .75 to reflect the full pocket depth.

80. Change ZSTEP to 0 to clean up the side walls of the pocket.

81. Pick CYCLE DATA and change the OFFSET VALUE to .25 for the finish path.

82. Select CREATE PATH and then COPY.

83. At the SEGMENT ID prompt type in 3 to indicate that the path to be copied is from Segment 3. The PATH COPIED message should appear.

As with all segments, you should now VERIFY Segment 4.

We are now done with the milling necessary for this job and will now begin to drill and tap the holes beginning with the 2, 1/2" holes.

84. Choose SEG ID and at the prompt, enter 5-DRILL.

85. The MILLING CYCLE (Y/N) prompt will appear and for the first time we will answer N (below the Y/N line) because this will be a drilling cycle, not a milling cycle.

86. Choose DRILL from the list of cycles that will appear.

87. Select SEGM PARAMS and pick TOOL INFO.

88. Pick TOOL# and change it to 2. This is the second tool we will use and we need to reflect that information.

89. Select EFF TOOL DIA and set it at .500.

90. Select Z DATA and pick ZDEPTH.

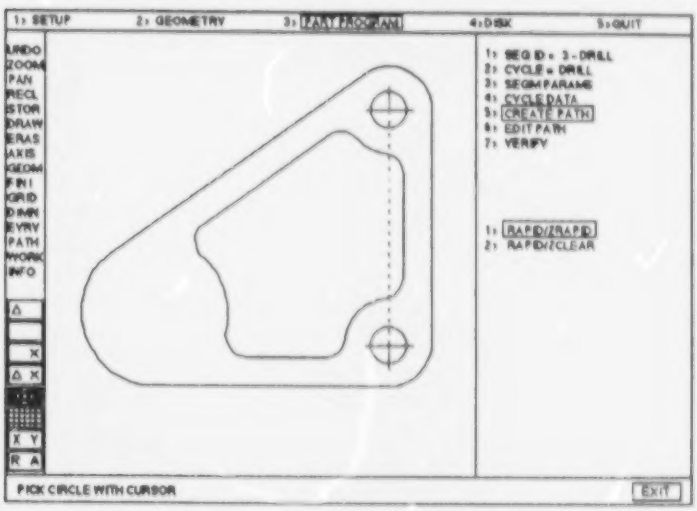
91. Set the ZDEPTH at 1.200. This number is arrived at by adding 1" for the part thickness plus 30% of the drill diameter (drill tip allowance) plus an overshoot of .050 to make sure that the hole breaks through completely.

92. Select CREATE PATH and choose DEFINE.

93. Select RAPID/ZRAPID this indicates rapid moves when moving from hole to hole (rapid moves are dashed blue lines) and rapid moves to and from the .05 clearance plane.

94. Now select the CENTER OF A CIRCLE icon. This one looks like a dot inside of a circle.

95. At the PICK CIRCLE WITH CURSOR prompt, move the cursor to the lower right .75 radius circle and click the mouse. A blue hole representing the drilled hole should appear. Do the same to the upper .75 radius circle. The existing tool paths should look like the following figure.



Now we will drill the eight tap drill holes.

96. Choose SEGMENT ID and enter 6-TAPDRILL at the prompt.

97. Say NO to the MILLING CYCLE? prompt and pick DRILL for the cycle.

98. Select SEGM PARAMS and choose TOOL INFO.

99. Select TOOL# and change it to 3.

100. Pick EFF TOOL DIA and set it at .201. This is the tap drill diameter for a 1/4-20 tap.

101. Pick Z DATA and then Z DEPTH. Set the Z DEPTH for .75. This is the depth of the tap drill holes on the drawing.

102. Pick CREATE PATH and then DEFINE.

103. Select RAPID/ZRAPID again.

104. Pick the RA icon. This will allow you to define a point on a specified radius at a specified angle on a circle whose center is at X0,Y0 only. Since our bolt circle is at X0,Y0 we can use this option. If it wasn't, we couldn't.

105. Enter .75 as the radius at the prompt.

106. Enter 90 for the angle. A .201 tap drill hole should appear at 90 degrees on the bolt circle.

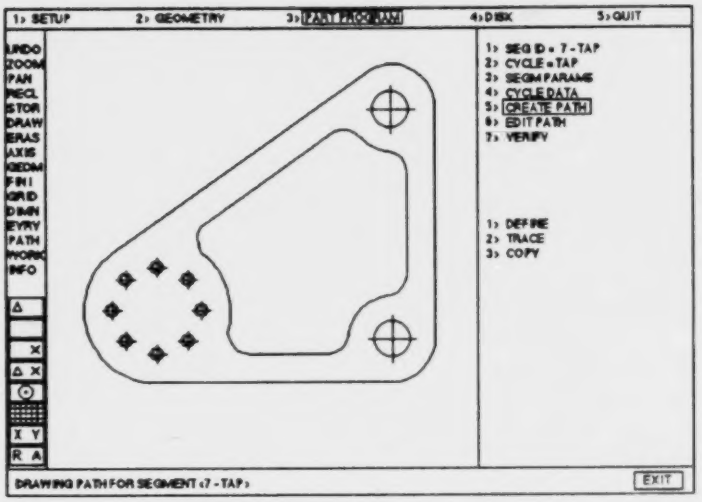
Now we will use the ability of the computer to repeat this hole 7 more times. This REPEAT option is very helpful and you should use it whenever possible.

107. Select EDIT PATH and choose REPEAT.

108. Choose ROTATE from the new menu.

109. Enter 45 for the ANGLE (the holes are 45 degrees apart) and 7 for the # OF COPIES (we have 1 hole already and want 7 more).

The tool paths should now look like this:



Now we will copy the tool path and tap the eight holes.

110. Choose SEG ID and enter 7-TAP at the prompt.

111. Say NO to MILLING CYCLE? and select TAP as the cycle.

112. Pick SEGM PARAMS and choose TOOL INFO.

113. Select TOOL# and make it 4.

114. Change the EFF TOOL DIA to .25. This is the diameter of the 1/4-20 tap.

115. Choose FEEDS & SPEEDS and set the SPINDLE SPEED at 200.

116. Select FEED (Z) and set it at 10. These last two settings are for a tapping head which will be mounted in the machine spindle.

117. Select Z DEPTH and set it at .5. This is for the 1/2 depth of the threaded hole called out in the drawing.

118. Select CREATE PATH and COPY.

119. Enter 6 after the SEGMENT ID prompt. The PATH COPIED message should appear. Since there is no way to verify this cycle visually, the following steps are necessary to "see" the tapped holes:

120. Pick ERAS and then pick EVRY from the concurrent commands.

You now have completed the steps necessary for the computer to write the program. Your blue tool paths should look like the part drawing minus dimensions. Now we will save the data on your disk.

121. Select DISK from the top menu in SIPP. Make sure that the DIR says A:\. If it doesn't, pick DIR and type in A:\ at the prompt.

122. Choose PART NAME and enter the name given to you by the instructor.

123. Choose SAVE and pick both of the choices in order. First, PART PROGRAM and after a message appears telling you that the program has been saved, pick GEOMETRY. You have now saved all of the data on your disk which is in the A: drive.

To load data from your disk:

124. Select DISK making sure that DIR says A:\.

125. Select LOAD and pick PART PROGRAM and then click on the blue ALL that will appear.

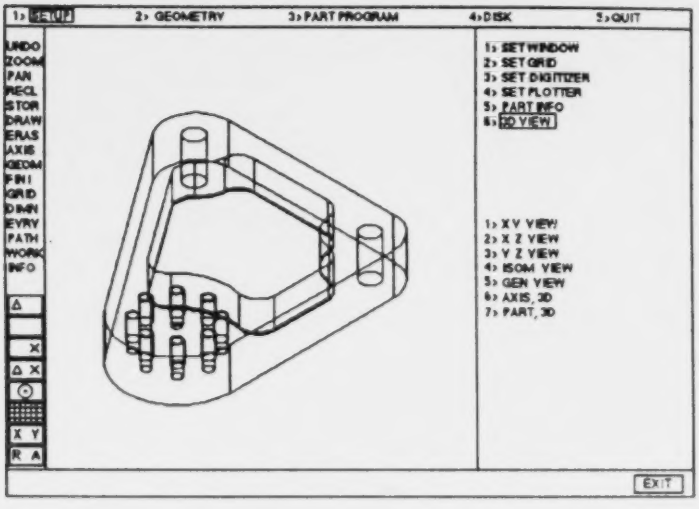
126. After the message says that the PART PROGRAM has been loaded, pick GEOMETRY.

Now we will create a 3-D view of the part.

127. When in SIPP, pick ERAS.

128. Choose SETUP and then 3D VIEW.

129. Select ISOM VIEW to indicate which 3D view you want. Nothing will happen at this point. Now pick PART 3D and the isometric view will appear. Many times, this view does not fit on the screen. If this happens: pick SET WINDOW and increase the SIZE. Then repeat steps 127-129. When the part appears to be on the screen correctly, EXIT and STOR to save the correct window size. A proper 3D view will look like this:



BEST COPY AVAILABLE

Now we print a copy of the 3D view. These steps must be followed to print any type of screen graphics.

130. Make sure that you have the picture the way that you want it to be printed. You cannot experiment with picture appearance during the printing process. Select ERAS.

131. Push the SHIFT and the 8 keys down at the same time.

132. Answer YES to the prompt.

133. Redraw the picture on the screen by following the 3D view display procedures. As the picture is being drawn on the screen, it is being sent to the printer. It is very important to make sure that the printer is turned on, that it has paper, and that your computer is connected to the correct printer. See your instructor if you have any questions about the printer.

134. Push the SHIFT and 8 keys down at the same time again.

135. Say YES to the second prompt. At this point the printer should begin to print the 3D view on the screen.

The only remaining step is to tell the computer to turn the data into a written CNC program.

136. Select QUIT from the top SIPP menu.

137. Pick EXIT SIPP. Note: Anything not saved will be lost permanently when you exit SIPP!!!

138. Answer YES to the CONFIRM...Y/N prompt.

This will take you to the main EZ MILL menu.

139. Select POST.

140. Make sure that the DIR says C:\EZ-CAM. If it doesn't, call up DIR and type it in exactly that way!

141. Choose CNC NAME and enter the name of the control you are using. See your instructor.

142. Answer YES to the DO YOU WANT TO LOAD prompt.

143. After the CNC INFO LOADED message appears, change the DIR back to A:\. Your disk is in the A: drive and we want to load your data from your disk.

144. Select PART NAME and enter the name you saved your data under.

145. Pick OUTPUT NAME and enter the same name as you did under PART NAME.

146. Select NC OPTIONS and answer all of the items on that menu according to your instructor. More information on the POST menu can be found in the User's Manual, Chapter 4. pages 1-8.

147. Pick SAVE OPTIONS and you will see the OPTIONS SAVED message.

148. After you make sure that the printer is ready, select RUN and answer YES to the ARE YOU SURE? prompt. At this point the system will translate all of the previous data into a written CNC program compatible with the type of control you have selected. See your instructor about loading that program into the CNC machine tool for machining.

149. After the program has been created, the following message will appear:

PART PROGRAM PROCESSED
HIT RETURN TO CONTINUE

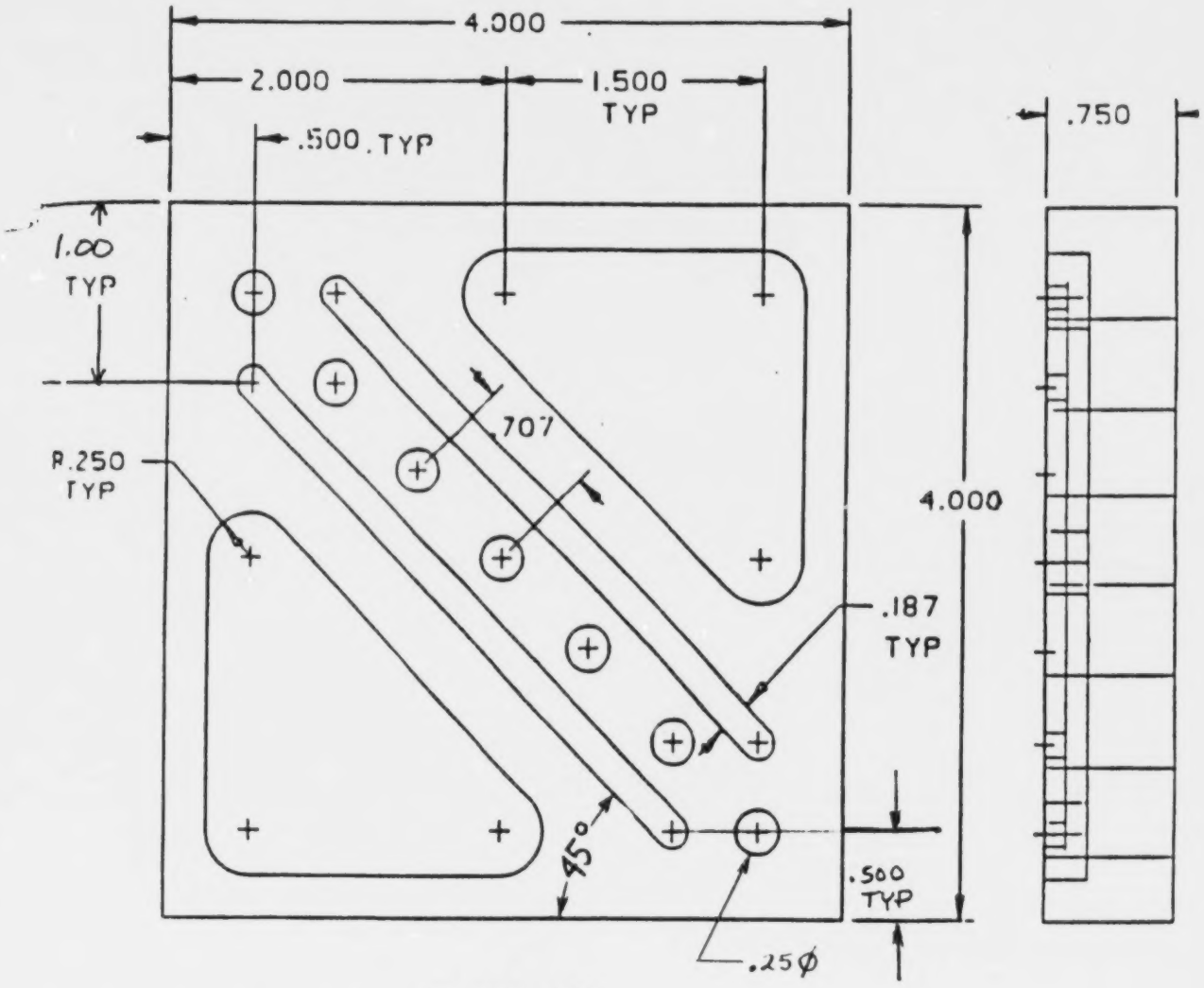
150. Hit the RETURN key to return to the POST menu. Then select QUIT from the POST menu and answer YES to the DO YOU WANT TO EXIT POST message. This will return you to the main EZ-MILL menu.

This exercise worked for one part. In each of the exercises you will do, different techniques will work. Becoming proficient at EZ-CAM involves exploring all of the menu choices available and picking the best one for each situation.

The software still revolves around creating geometry, developing part programs, saving on the disk, and posting the program.

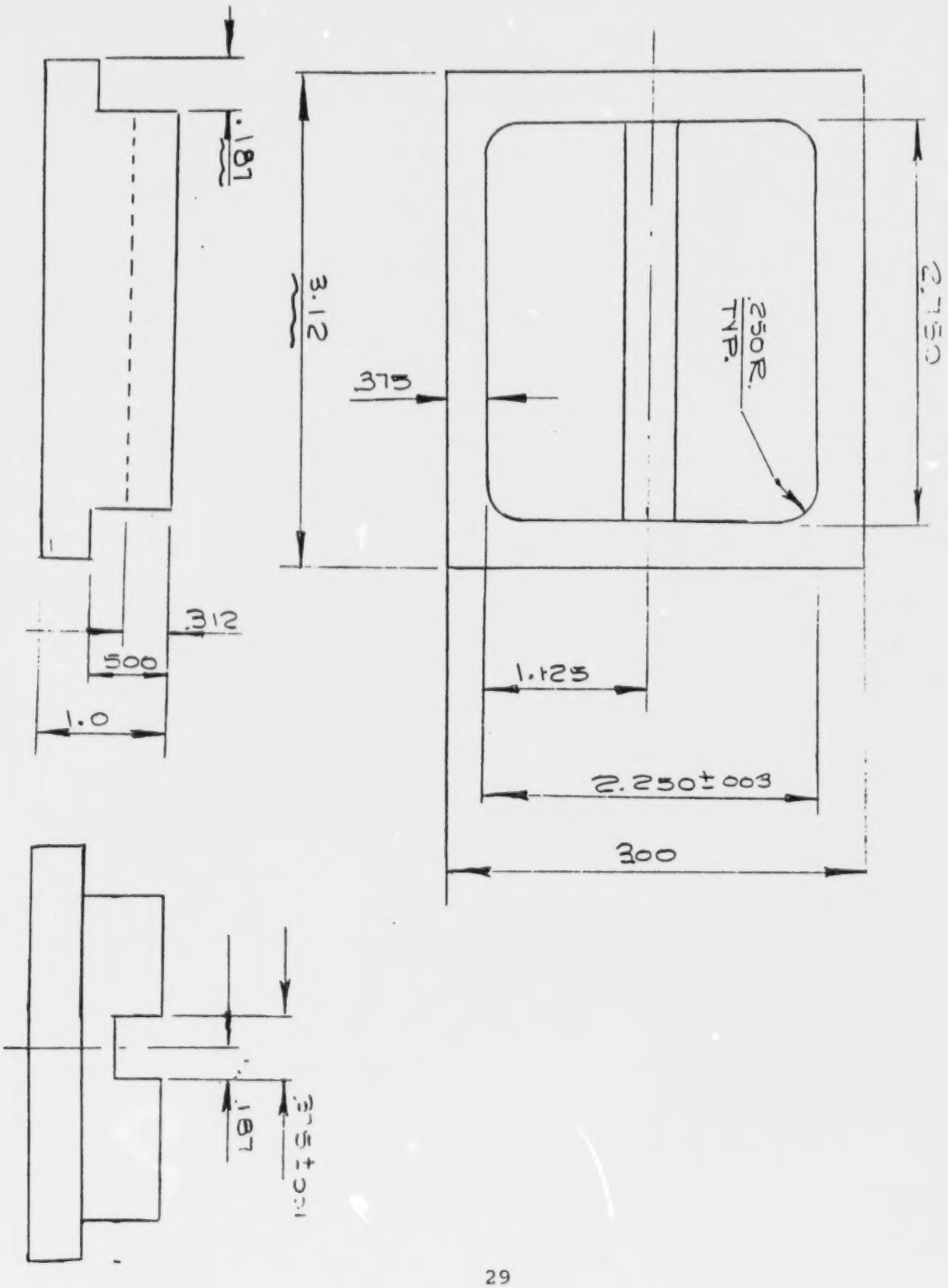
ASSIGNMENT UNIT 1

Using the EZ-CAM, EZ-MILL software, create CNC programs for the following projects. Submit a printed 3D view and a written CNC program for each job. Consult with the instructor regarding specific instructions for each project.

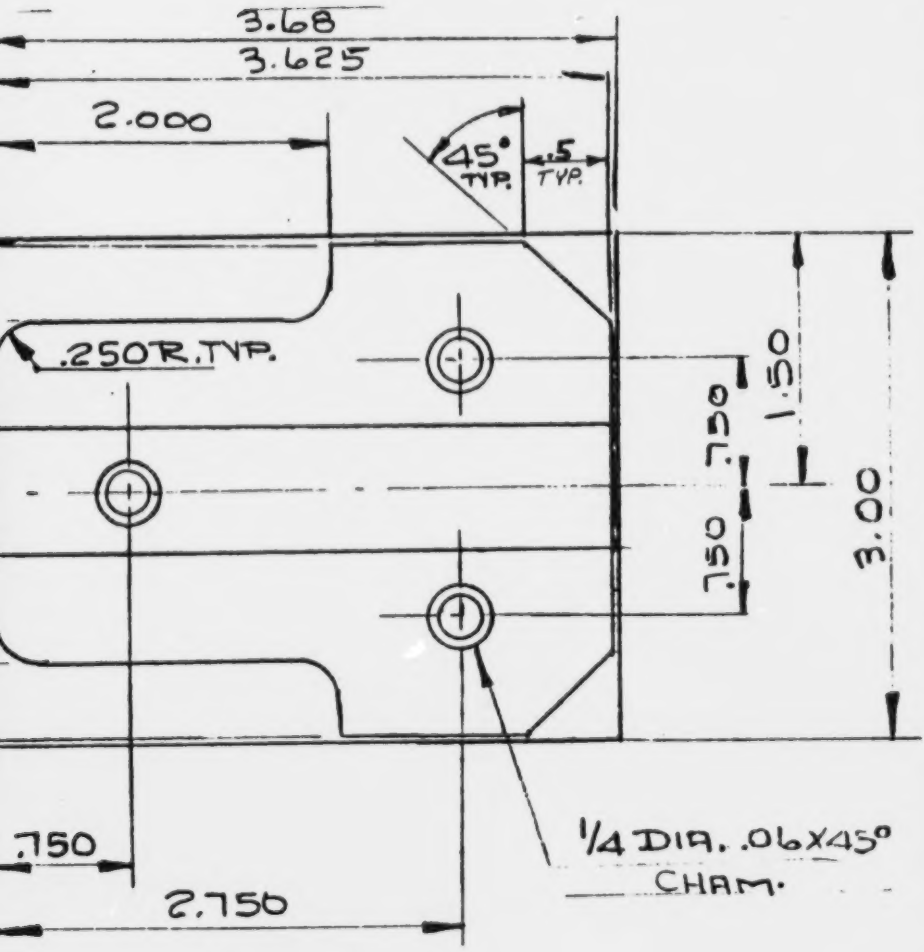


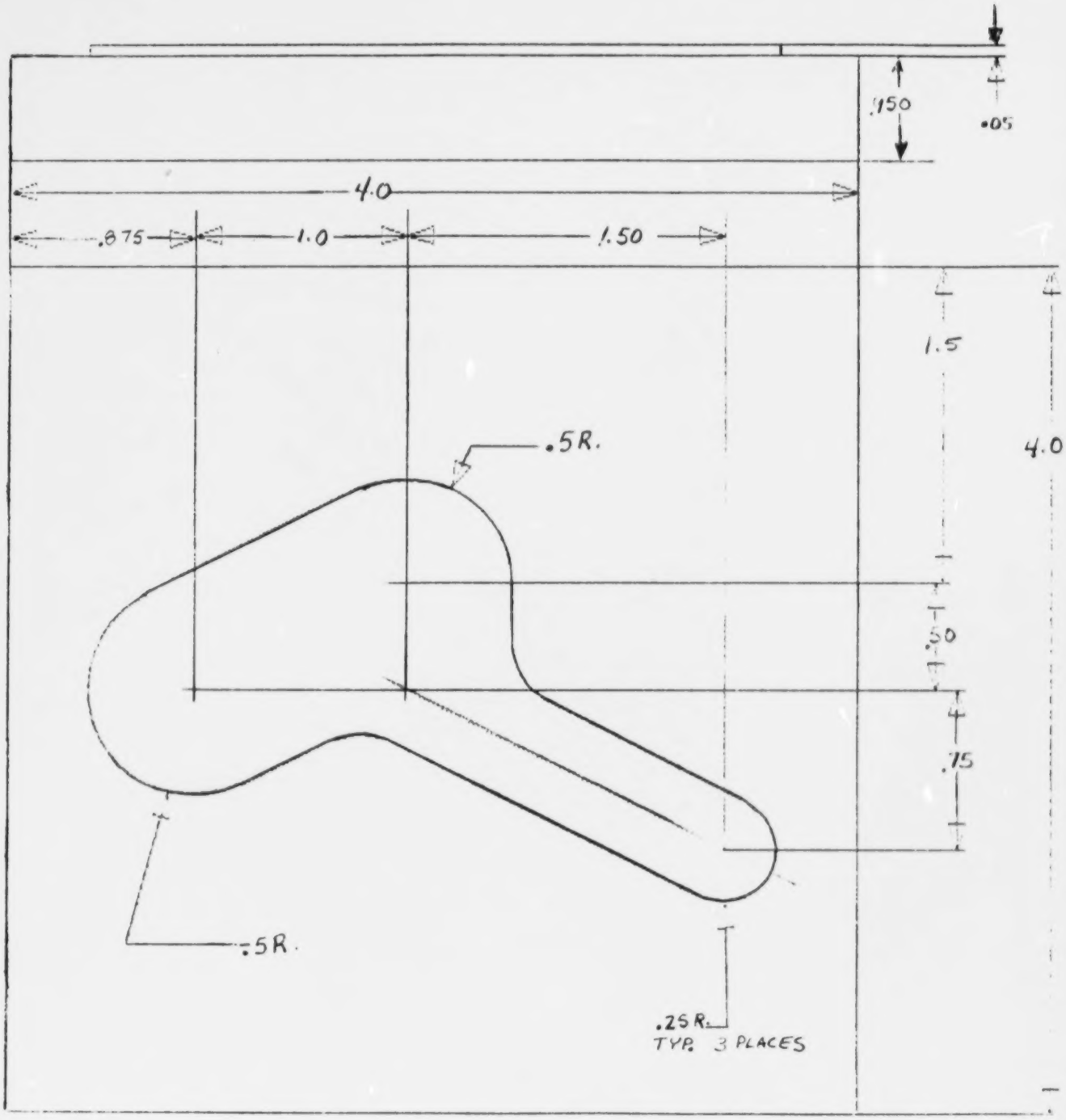
SLOTS ARE
.125 DEEP

POCKETS ARE
.250 DEEP


.187
3.12
.375
.250R.
TYP.
2.750
.312
.500
1.0
1.125
2.250±.003
3.00
.187
1.0±.015

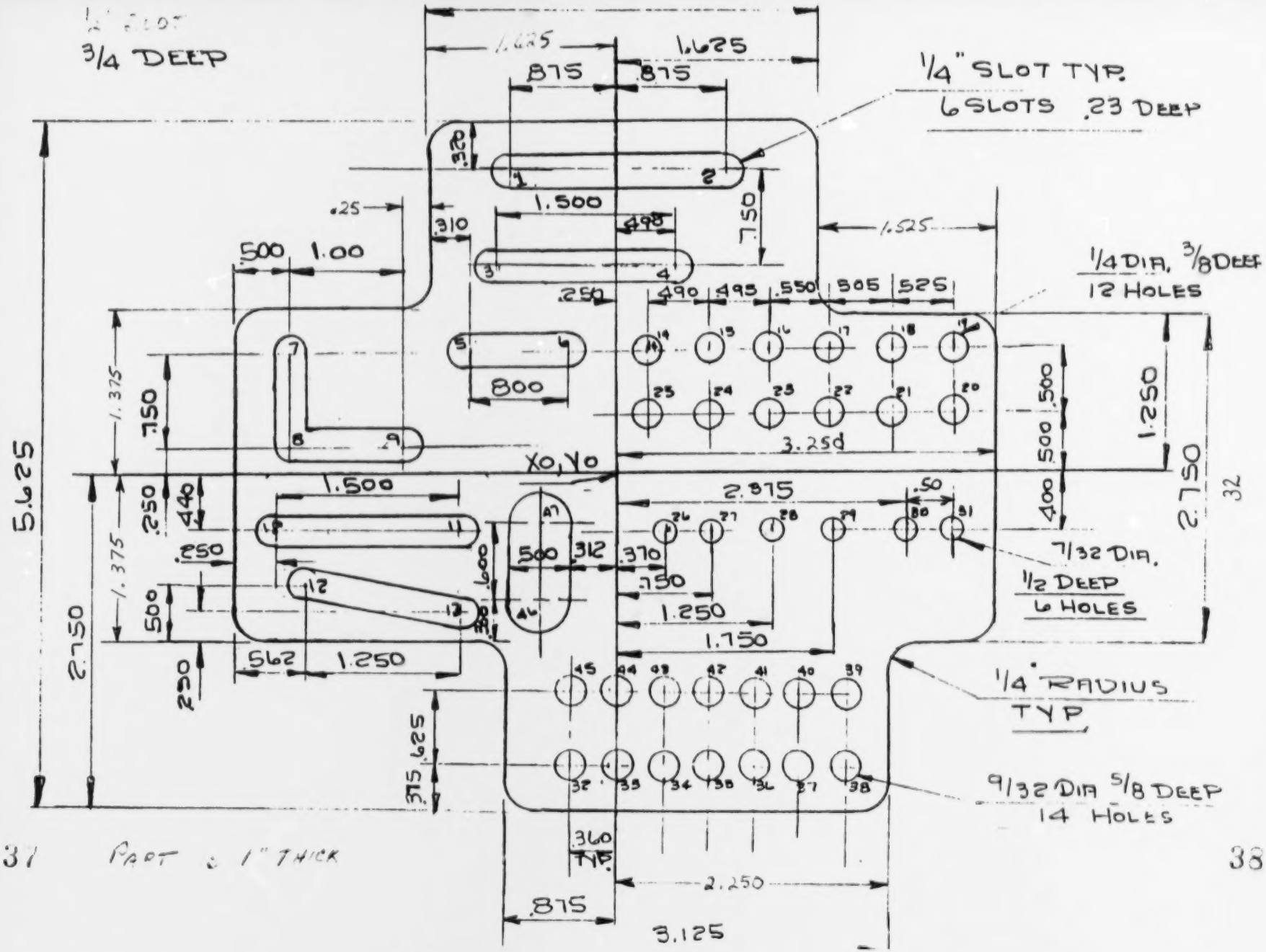
.375
.312
.750
.02
2.960
2.000
1.480
1.000
.375
.750


3.68
3.625
2.000
45° TYP.
.5 TYP.
.250R. TYP.
1.50
.750
.750
3.00
.750
2.750
1/4 DIA. .06X45° CHAM.



(ISLAND MILL) ASK INSTRUCTOR FOR HAND OUT SHEET

BEST COPY AVAILABLE


1/4" SLOT
3/4 DEEP
1.625
1.625
.875
.875
1/4" SLOT TYP.
6 SLOTS .23 DEEP
.320
1.500
.498
.750
.25
.310
.500
1.00
1.525
1/4 DIA. 3/8 DEEP
12 HOLES
.250
.490
.495
.550
.505
.525
.800
1.375
.750
5.625
3.250
.500
.500
.400
1.250
2.750
X0,Y0
1.500
2.875
.50
.250
.440
.250
.500
.312
.370
.750
1.250
1.750
7/32 DIA.
1/2 DEEP
6 HOLES
.600
.562
1.250
.250
2.750
1/4 RADIUS
TYP
.375
.625
9/32 DIA 5/8 DEEP
14 HOLES
.360
TYP.
2.250
.875
3.125
31
PART = 1" THICK
32
38

MILWAUKEE AREA TECHNICAL COLLEGE

ADVANCED CNC PROGRAMMING FOR JOURNEYMEN

Unit 2 EZ-EDM

INTRODUCTION

In this unit you will learn how to use computer software to describe a part and to use that description to generate a written CNC program. In this course we will use the Bridgeport EZ-CAM software. In this unit we will a portion of the EZ-CAM software which is devoted to EDM (Electrical Discharge Machining), EZ-EDM.

INSTRUCTIONAL OBJECTIVES

When you have completed this unit you will be able to:

1. Correctly enter and exit EZ-CAM software.
2. Correctly use the EZ-EDM software to create a CNC program.
3. Properly use the computer and printer to print out a toolpath.
4. Properly use the computer and printer to print out a CNC program.

LEARNING MATERIALS

1. Advanced CNC Programming for Journeymen, Information and Assignment Unit 2.
2. EZ-EDM User's Manual, Chapters 1-4.

LEARNING ACTIVITIES

1. Read Advanced CNC Programming for Journeymen, Information Unit 2. (Objectives 1-4)
2. Read EZ-EDM User's Manual, Chapters 1-4. (Objectives 1-4)
3. Complete Advanced CNC Programming for Journeymen, Assignment Unit 2. (Objectives 1-4)

ASSIGNMENT

1. Read Information Unit 2.
2. Attend the Lecture/Discussions on Unit 2.
3. Read Chapters 1-4 from the EZ-EDM User's Manual.
4. Complete Assignment Unit 2.

INFORMATION UNIT 2

EZ-EDM is a software package designed to create part programs for EDM machines. It is similar to EZ-Mill in the way that it does this. The menus are basically the same and the same procedures are followed. In this job we will follow this procedure:

1. We will determine where the part 0,0 point will be according to the drawing.

2. We will set the window size and location in SETUP.

3. We will create the part geometry in GEOMETRY.

4. We will define the cycle parameters in PART PROGRAM. It is in this area that EZ-EDM differs from EZ-Mill. The parameters for a EDM machine are different than those for a milling machine. A milling machine uses a cutter, a wire EDM uses a wire, etc.

5. We will create the path in PART PROGRAM.

6. We will save the data in DISK.

7. We will create a CNC program in POST.

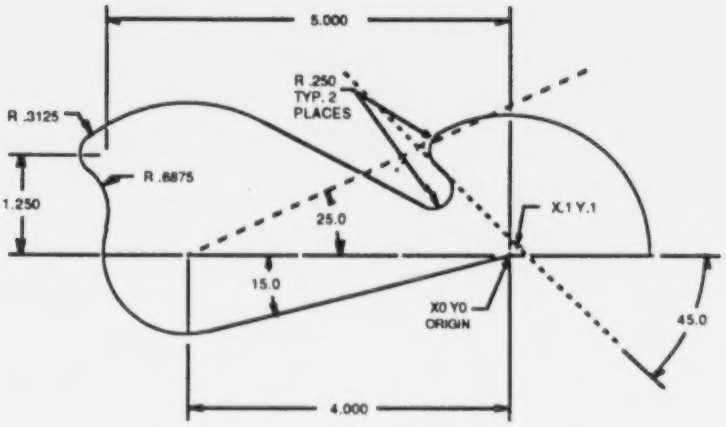
We must first review the steps for entering the software.

1. When the T305 Software Menu is on the screen, type EZ to enter EZ-Cam.

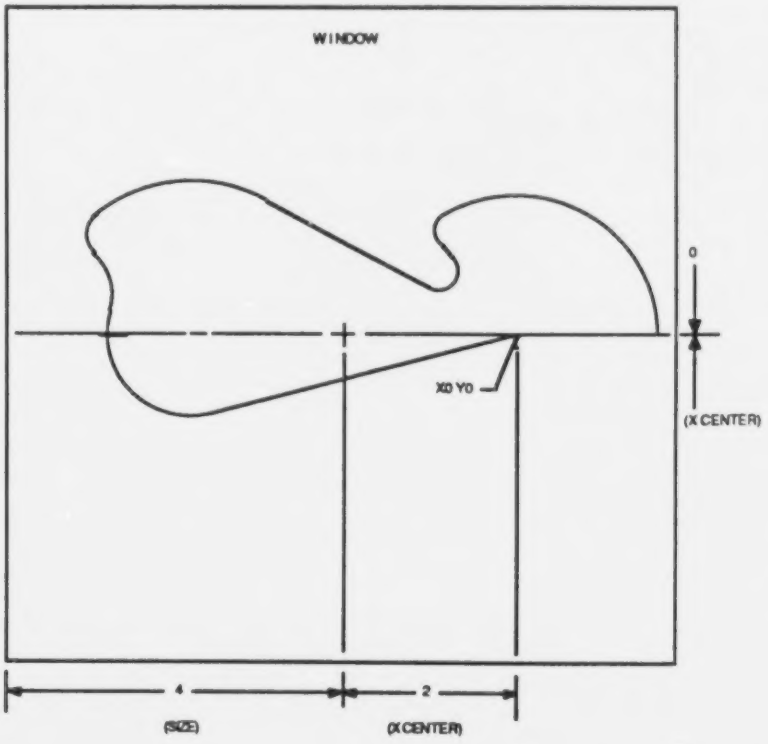
2. When the Windows Software comes on the screen, move the cursor with the mouse to EZ-EDM.bat and click the left mouse button twice rapidly. The hourglass symbol will appear and then you will see the EZ-EDM main menu.

Refer to the drawing on the following page for this lesson. We will be wire EDMing out this die block opening in a piece that is 1" thick. There will be a 2 degree tapered clearance for blanking through that is .800 thick and a .200 straight die life wall..

The first step is to determine part 0,0 and to use that in creating the window. The following figure illustrates where part 0,0 was set for this job.



Using the techniques learned in the previous lesson and following the next picture as an example, set the window to the following parameters: X CENTER -2, Y CENTER 0, SIZE 4



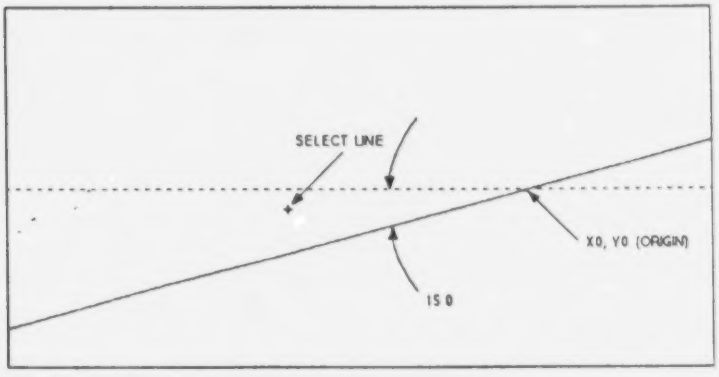
Once you have set the window EXIT to see the shift occur and if it is correct then pick STOR from the concurrent commands and say YES.

Go to GEOMETRY and the set the MODE to DEFINE if it is not there already. We will define the geometry for this die by going through the following steps:

1. Choose LINE and then the PARAL X AXIS option. Once again, this lesson will be using only the menu choices necessary for this job. There are many others which will be more appropriate for other jobs. Chapter 3 in the EZ-EDM User's Manual explains these other menu items in detail.

2. Enter an offset value of 0.

3. Select LINE and then the PT,ANGLE,LINE option. This will define a line through a specified point at a specified angle to a specified existing line. You must select the existing line on the inside of the angle formed by it and the new line. Refer to the following figure.



4. Enter 15 for the ANGLE.

5. Select the XY icon from the icon menu on the lower left side of the screen.

6. Enter 0 for X LOCATION and 0 for the Y LOCATION. We have now told the computer to create a line, at 15 degrees from the horizontal line, which goes through a point at X0,Y0.

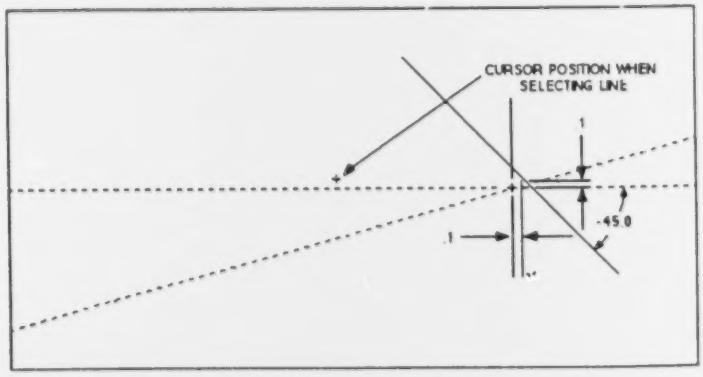
7. Move the cursor to the X axis as shown in the preceding figure and click. The angled line should appear. If at any time something appears in the wrong place, select UNDO from the concurrent commands to erase the mistake.

8. Choose LINE and again the PT,ANGLE,LINE option.

9. Enter -45.

10. Select the XY icon and enter .1 for X and .1 for Y.

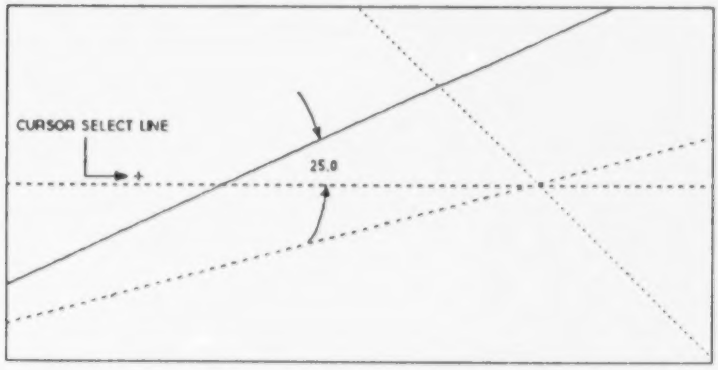
11. Move the cursor above the X-axis line as shown in the following figure. When you click, the 45 degree line should appear and your screen should look like this figure.



12. Select LINE and PT,ANGLE,LINE option.

13. Enter 25 for the angle.

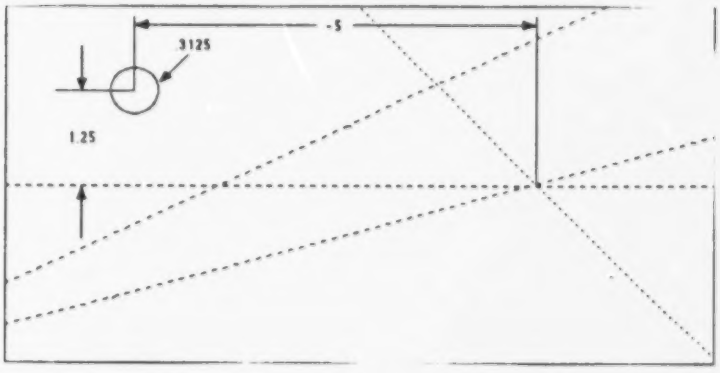
14. Select the XY icon and enter -4 for X and 0 for Y. Move the cursor to slightly above the X-axis and click. The following figure will illustrate cursor position and how your screen should look.



15. Select CIRCLE and the CENTER COORD option.

16. Enter .3125 as the RADIUS.

17. Enter -5 for X center and 1.25 as Y center. The following circle should appear.



18. Pick CIRCLE,R=? as an option. This will allow you to define a circle of an unknown radius providing certain other information is known.

19. Choose CENTER,TANG L. This will define a circle of an unknown radius about a specified point and tangent to a specified line.

20. Select the bottom line (Line 2 in the following figure). This gives us the tangency line.

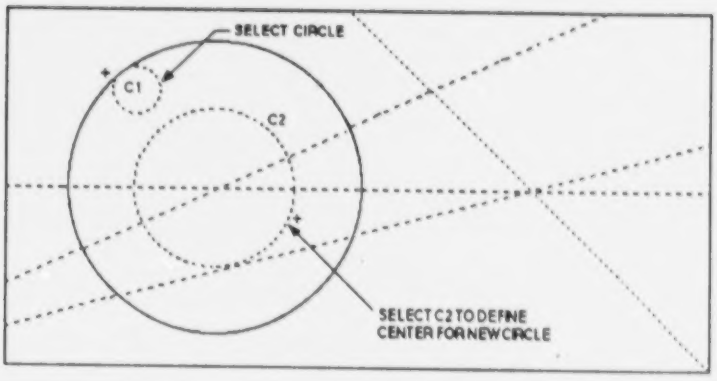
22. Pick the X icon and select the intersection of lines 1 and 4. This gives us our center point. Use the following figure as an example. After you have clicked on the intersection point, a circle will appear and your screen should look like this:

23. Stay in CIRCLE,R=? and select option CEN,TANG C;B. This defines a Big circle with a specified center point which surrounds and is tangent to a smaller circle

24. Since the large circle will have the same intersection of Lines 1 and 4 for its center as did the previous circle (C2) we do not have to redefine that center point.

25. Move the cursor to Circle C1 and click on the upper-left corner as illustrated in the following figure.

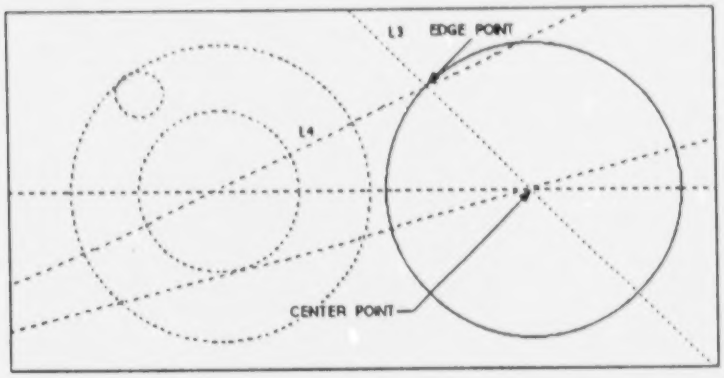
26. Select the CENTER OF A CIRCLE icon (it looks like a bullseye) and click on Circle C2. The large circle will appear and the screen should look like the following figure.



27. Stay in CIRCLE,R=? and pick the POINT, CENTER option.

28. Select the X icon and choose the intersection of lines 3 and 4.

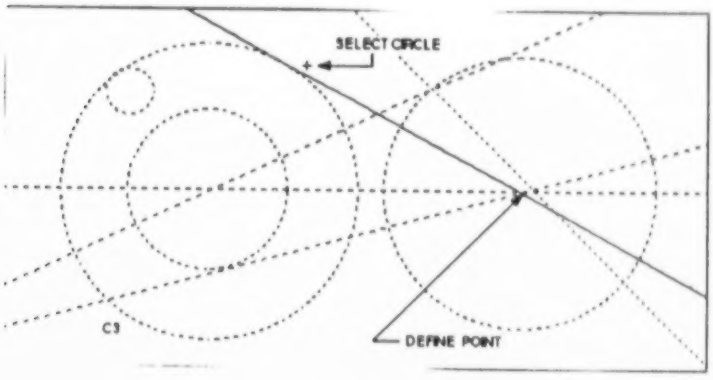
29. Select the XY icon and enter 0 for X location and 0 for Y location. A circle will be drawn with a 0,0 center point which goes through the specified intersection point. The screen should look like this:



30. Select LINE and the PT,TANGENT C option. This will create a line which goes through a specified point tangent to a specified circle.

31. Select the XY icon and enter 0 for X location and 0 for Y location.

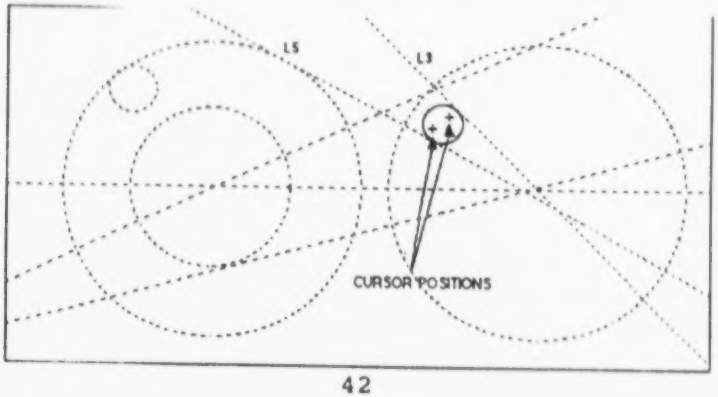
32. Move the cursor to approximately 1 o'clock on the large circle (C3) and click. A line should appear which is tangent to Circle 3 and which goes through the 0,0 point. The screen should now look like this:



33. Select CIRCLE and the TANGENT L,L option. This will define a circle of a specified radius which is tangent to 2 specified lines.

34. Enter .25 for the RADIUS.

35. In response to the PICK LINE WITH CURSOR prompt, select line 3 and then line 5 <u>on the inside of the angle formed by these two lines as shown in the following figure.</u> The screen should now look like this:

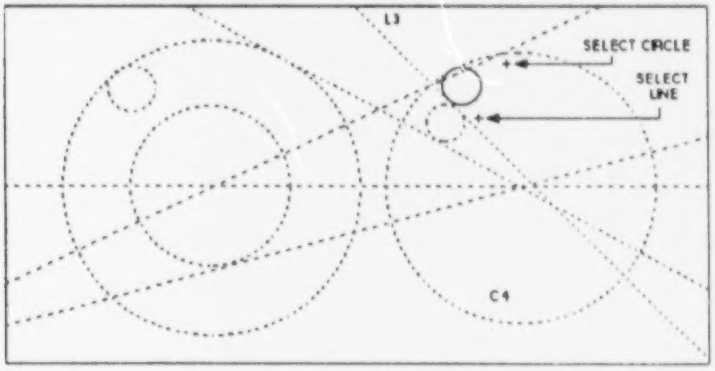


36. Stay in CIRCLE and select TANGENT L,C as the option. This will define a circle tangent to a specified line and to a specified circle.

37. Enter .25 for the RADIUS.

38. In response to the PICK LINE WITH CURSOR prompt, click on right of line 3 as shown in the following figure.

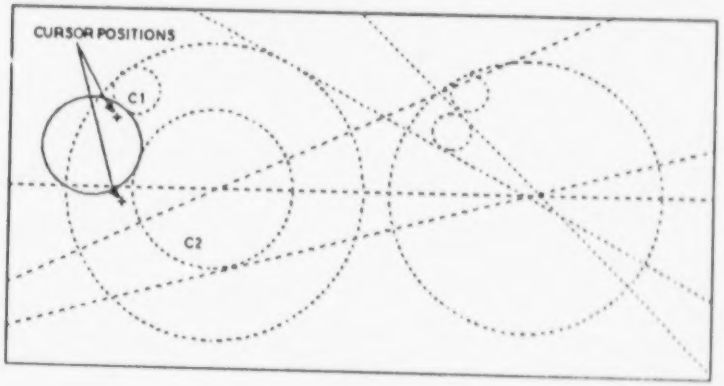
39. In response to the PICK CIRCLE WITH CURSOR prompt, click on Circle 4 at about 12 o'clock as shown in the following figure. The screen should now look like this:



40. Stay in CIRCLE and select TANGENT C,C from that menu. This will define a circle which is tangent to 2 other specified circles.

41. Enter .6875 for the RADIUS.

42. Click on the lower left outside of Circle 1 and on the left outside of Circle 2 as shown in the following figure. The screen should now look like this:



BEST COPY AVAILABLE

We have now defined all of the geometry necessary to execute this part. This unit illustrates another very valuable feature of this software. Not only does it write the actual CNC program, line by line, and thus eliminates much of that tedium, it also allows us to create geometry with very little information. If you return to the drawing on page 36 it appears as though there is not enough information there (without a lot of trigonometric calculations) to define the geometry. But by using some of the more sophisticated menu choices, we allow the computer to work for us.

Up to now, EZ-EDM seems to be just like EZ-Mill. That is because they both define Windows and Geometry the same way. But as we begin to create the Part Program we will see that they are truly different.

We will create the Part Program using the following steps:

43. Select PART PROGRAM from the Top Menu.

44. After the SEG ID prompt, type and enter 1-TAPER. Just as in EZ-Mill, EZ-EDM also requires that Segment ID's be a number followed by a dash followed by a short descriptive phrase. This format is very important.

45. The CYCLE menu will appear. Notice that this menu is different from the one that appears at this point in EZ-Mill. Milling and wire EDM are very different machining processes. It is natural that they would require answers to different types of questions. These menu choices are explained further in the EZ-EDM User's Manual, Chapter 3. Select CONTOURING from this menu.

46. Choose SEGM PARAMS and then POSITION & FEED.

47. Pick UPPER ARM POSITION. This allows you to tell the computer, in millimeters, how high the upper arm of the wire guide is. This setting will be obtained from setting the wire EDM up with the workpiece in place and then reading the upper arm height setting from the machine. For this job type and enter 25 (25mm).

48. Select AD TABLE HT. Additional table height is the distance above the surface of the work table where the dimensions must be held. It is the point where the wire will be tilted for burning a taper. Since we have a 1" thick piece with a .200" land, the taper will begin at .800" above the table top. Type and enter .800.

49. Pick CREATE PATH and then DEFINE.

50. Select the ARC C-CW option. Use this choice when the tool path will be moving about an arc in a counterclockwise direction.

51. Before we can begin to move through the geometry (putting blue tool paths on top of green geometry) we must define the type of burn we want using a series of menu choices called the MOTION RECORD.

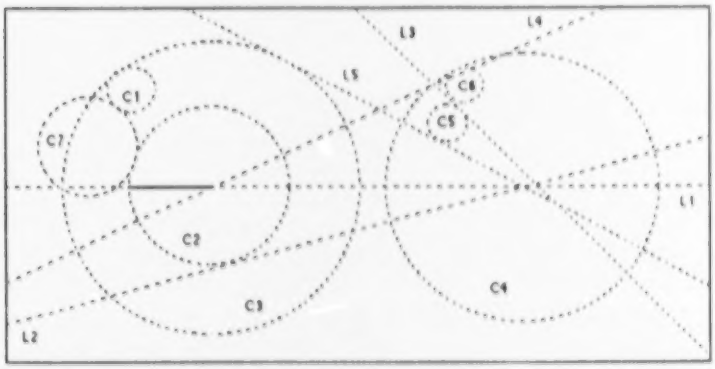
52. Pick TAPER and choose a TAPER STATUS of NORMAL. NORMAL taper maintains a constant taper throughout the burn.

53. Enter a TAPER ANGLE of 2. This will cause a constant taper of 2 degrees per side throughout the burn.

54. Select COMP and set COMP STATUS to left. This will move the wire to the left of the tool path <u>as it move through the job in the direction you establish by the order in which the blue lines are created.</u>

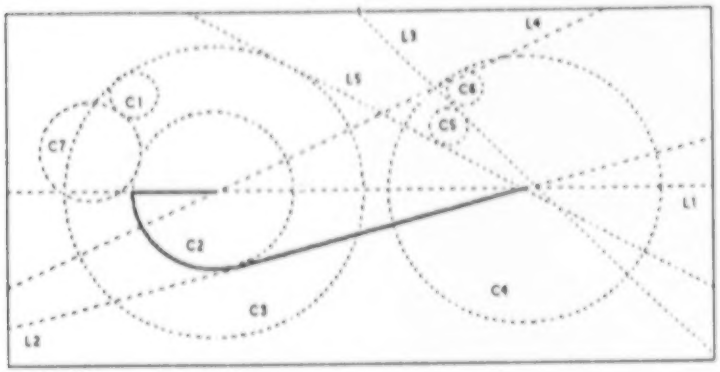
55. Enter a COMP VALUE of .0085. This number reflects the radius of a .010" diameter wire (.005") plus an <u>estimated</u> spark gap of .0035". The size of the spark gap is determine by the power settings on the wire EDM. This number can only be estimated unless many hours of experience has provided empirical examples to use. This setting completes the MOTION RECORD.

56. Select the X icon and begin to move through the geometry creating blue tool paths. Begin with the intersection of Lines 1 & 4. Move to the left along Line 1 and select the intersection of Line 1 and Circle 2 as shown in the following figure.



57. Now click on the intersection of Line 2 and Circle 2.

58. Pick the intersection of Lines 1 & 2 next. The tool path should look like this now.



59. Proceed to the far right and pick the intersection of Line 1 and Circle 4.

60. Pick the intersection of Circles 4 & 6. At this point the geometry gets pretty small. It will be helpful to pick ZOOM from the concurrent commands and "drag" a box around Circles 5 & 6 while holding the mouse button down, releasing the button when the box is correct. RECL from the concurrent commands will restore a ZOOM to the stored window size.

61. Select the intersection of Circle 6 and Line 3.

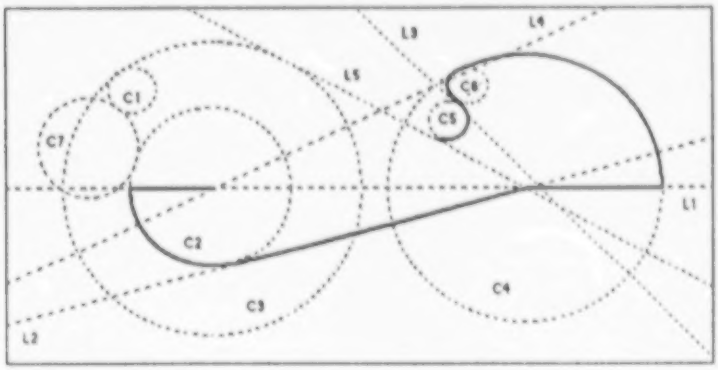
62. Now pick the intersection of Line 3 and Circle 5.

63. Change the Mode to ARC TANGENT. This works well with lines tangent to circles.

64. Select the point where Circle 5 and Line 5 intersect.

65. Pick the intersection of Line 5 and Circle 3.

66. Move to the intersection of Circles 3 & 1. The tool path should now look like this:

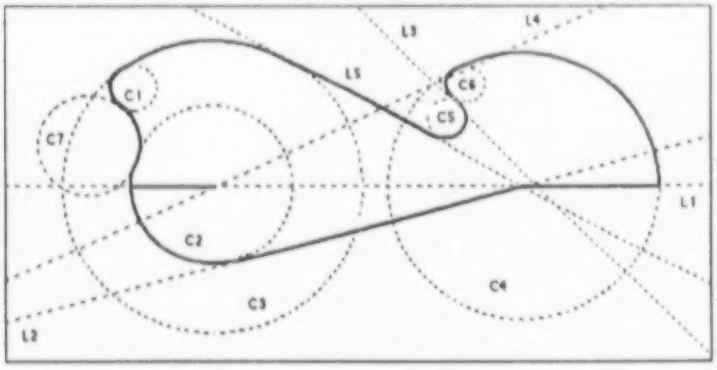


67. Move to the intersection of Circles 1 & 7.

68. Pick the intersection of Circles 7 & 2.

69. Change the icon to the Point icon (it looks like a small triangle). Paths should always end on a point.

70. To complete the path, select the intersection of Circle 2 and Line 1. The completed tool path should look like this. Compare it with the part drawing at the beginning of this unit.



We have just completed the tool path which will burn a 2 degree/side tapered opening which will follow our specified contour. We now need to straighten out the wire and burn the .200" die life land in about the same contour. We do that this way:

71. Select TAPER from the Motion Record and turn it the TAPER STATUS OFF. We don't need the Taper on any longer.

72. Select COMP and turn the COMP STATUS OFF.

73. Open a second segment by picking SEG ID and entering 2-LAND at the prompt.

74. Pick CONTOURING as the cycle.

75. Erase the screen and redraw the tool path by picking ERAS and EVRY from the concurrent commands.

76. Go into CREATE PATH and COPY. Enter 1 at the prompt. The PATH COPIED message should appear.

77. Select VERIFY and then MOTION RECORD.

78. Pick GLOBAL CHANGE. This will change all of the settings in a category for this segment. Set the TAPER STATUS to OFF.

This completes the creating of the data necessary for the CNC program. Anytime you are done or exit SIPP you should save your work. Do this by calling up DISK and then:

79. Make sure that the DIR is A:\. If it isn't, call up DIR and type A:\ in.

80. Pick PART NAME and enter the name you wish to give to the part.

81. Choose SAVE and then PART PROGRAM

82. After the PART PROGRAM SAVED message, choose GEOMETRY.

You cannot display a 3D view of wire EDM paths. In order to print out a view of the tool path:

83. Push the SHIFT key and the 8 key down at the same time.

84. Click on the YES.

85. Erase the screen and draw the tool path by picking ERAS and EVRY from the concurrent commands.

86. Push the SHIFT and 8 down together again and say YES again. If you are connected to a printer which is working, you should get a printout of the tool path at this point.

To Post the program:

87. Exit SIPP and go to POST.

88. Change the DIR to C:\EZ-CAM if it is not that.

89. Choose CNC name and enter JPXTIA. This stands for the JaPaX Taper Inch Absolute post. This is the one we are using for this job. Consult with the instructor regarding the post for other jobs.

90. Say YES to the DO YOU WANT TO LOAD JPXTIA? message.

91. Change the DIR to A:\. This is so that you can load your data from the disk which is in the A:\ drive.

92. Select PART NAME and enter the name that you save the job as. Give the same name for OUTPUT NAME.

93. Select NC OPTIONS and set them according to your instructor.

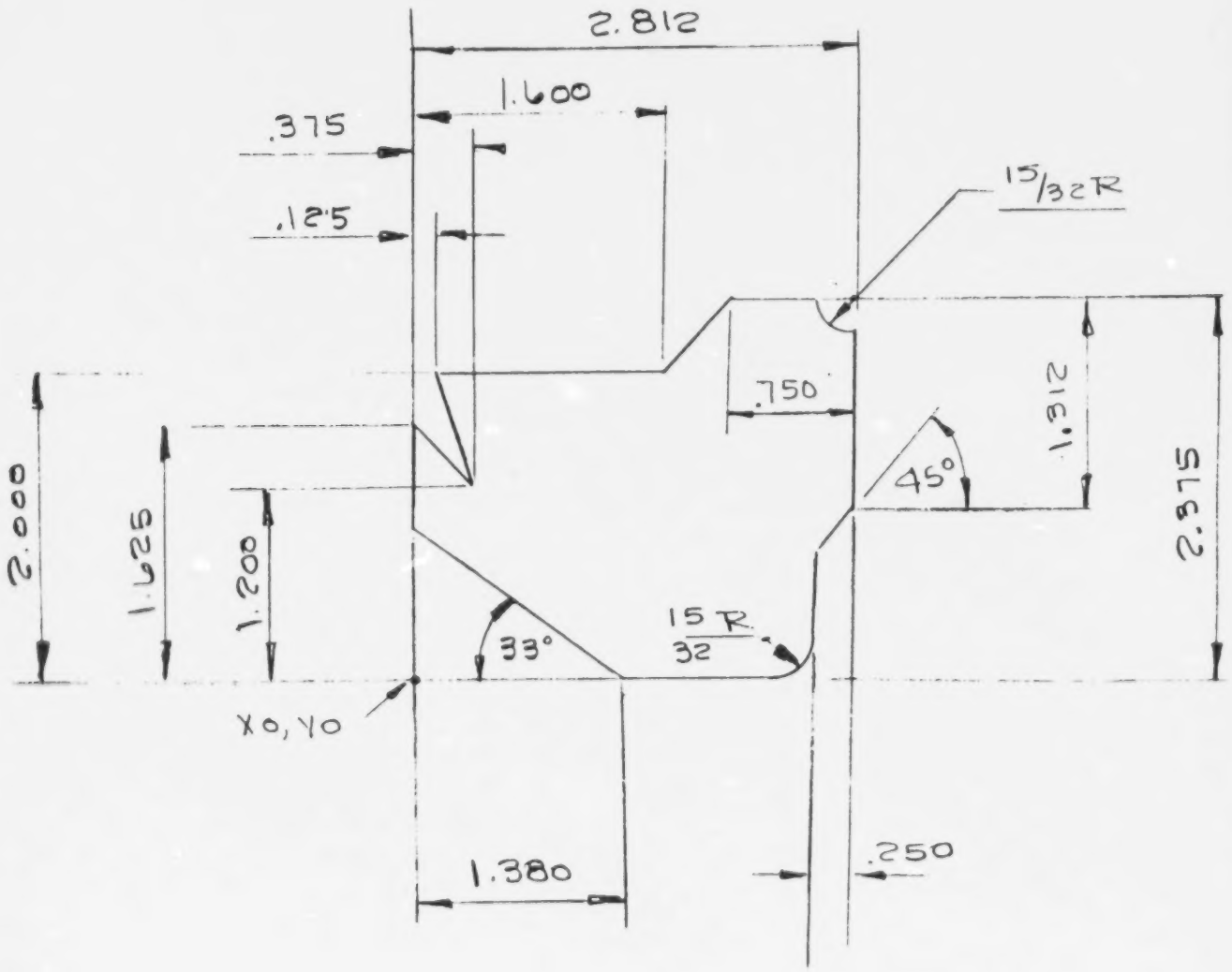
94. Pick SAVE OPTIONS and save them once they have been set.

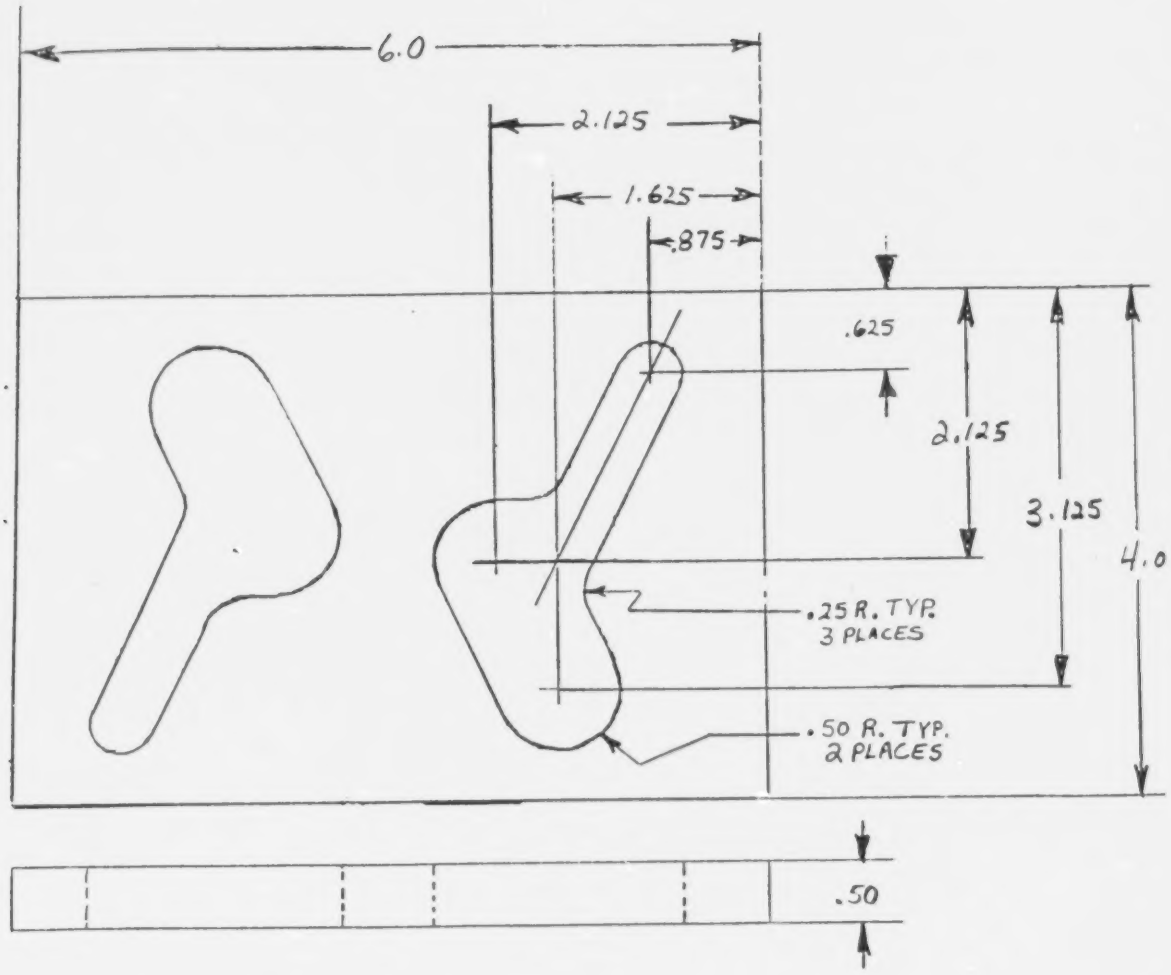
95. Select RUN and say YES to the ARE YOU SURE? message.

At this point the program should be created and this job is done.

ASSIGNMENT UNIT 2

Use the EZ-EDM software to create a printout of the tool path and a written CNC program for the following die openings. Consult with your instructor regarding all machining settings.




6.0
2.125
1.625
.875
.625
2.125
3.125
4.0
.25 R. TYP.
3 PLACES
.50 R. TYP.
2 PLACES
.50

51